



CNC Routing Overview (9 March 2019)
J.Sikorski (john2pt0)

| | |
|--|----|
| Objectives | 2 |
| Safety | 2 |
| Computer Numeric Controlled (CNC) Machining | 3 |
| Taking a project from design to machine ready | 6 |
| Feeds & Speeds in the Woodshop | 6 |
| CAD / CAM (short intro. Much more info is on the Vectric website.) | 9 |
| Postprocessing | 17 |
| Set Up and Run Your Project | 18 |
| Run Your Project (zero to the mill Z0 each time you change mills) | 21 |
| What could possibly go wrong? | 22 |
| Acknowledgements | 24 |
| References | 24 |
| End mill info (no endorsement by our facility or guarantee of quality / results) | 24 |

Objectives

This document will introduce you to:

- a) the **safety precautions** for using the CNC router
- b) the **terminology** of CNC machining (**CAD, CAM, postprocessing**)
- c) **taking a project from design to machine ready** (GWizard, VCarve)
- d) the **setup and running** of a project (**Mach 3**)

This is not a substitute for taking the CNC course.

You will need to take the course to get our makerspace VCarve registration number to use VCarve with our CNC router.

To be **certified** to use the Hive 1 3 CNC router, you will need to take the Hive 1 3 CNC Intro Course and then arrange for a Hive 1 3 **CNC machinist** to observe you running a project. Post your proposed time & date on the hive13.org website mailing list page to ask for coverage.

Safety

Protect yourself:

- safety glasses, ear protection,**
- no hair, lanyards or hoodie ties dangling out it front**
 - tie up and tuck in the back of your shirt**
- no gloves!**
- closed-toe shoes

Protect your colleagues:

- Secure the mill (cutter).
- Secure the material.
- Avoid loose wood with loose knots or cracks.

Protect the machine:

- Secure the mill.
- Secure the material:
 - Use nails at least 1/2" longer than the thickness of your material.
 - Nail every 6-12", including along cuts along the center of the board.
 - If using mills > 3/8" in diameter, consider using screws as holddowns.

Don't cut more than 0.04" (1mm) into the spoil board

(0.02" should suffice for MDF, plywood, plastic and aluminium)

Do Not Leave the Machine While It's Running!

Computer Numeric Controlled (CNC) Machining

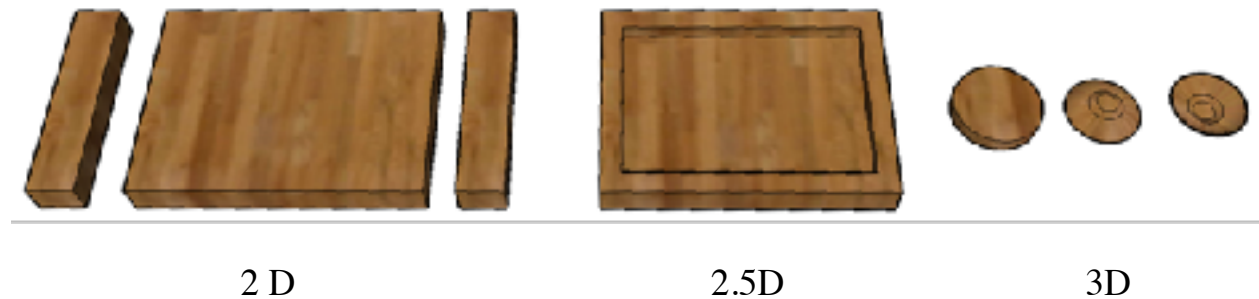
CNC machining is “subtractive” (like wood carving) and can cut 2D, 2.5D, and 3D tool paths.

2D - cuts in one plane (like cutting a board to size on the table saw or cutting holes with the jig saw)

2.5D - moves the mill to different levels and then does 2D cutting at that level to form holes or pockets.

(like using a hand held router to cut a serving tray from a single board; you level the entire top of the 12” x 8” board to set the thickness, then level the middle area of the board 1/2” lower, leaving edges elevated 1/2” along each side.)

3D - cutting objects smoothly rounded on one or both sides (like shaping a saucer, as one can on the lathe but here you lay the wood flat to smooth one side, then flip it over to smooth the other)



CNC work include 3 stages:

CAD (computer aided design) - draw up the part in the software of your choice

VCarve, Illustrator, Sketchup, Fusion 360, Inkscape, etc.

CAM (computer aided manufacturing) - decide what mills will cut which features, that is, define the toolpaths in software that provides this function

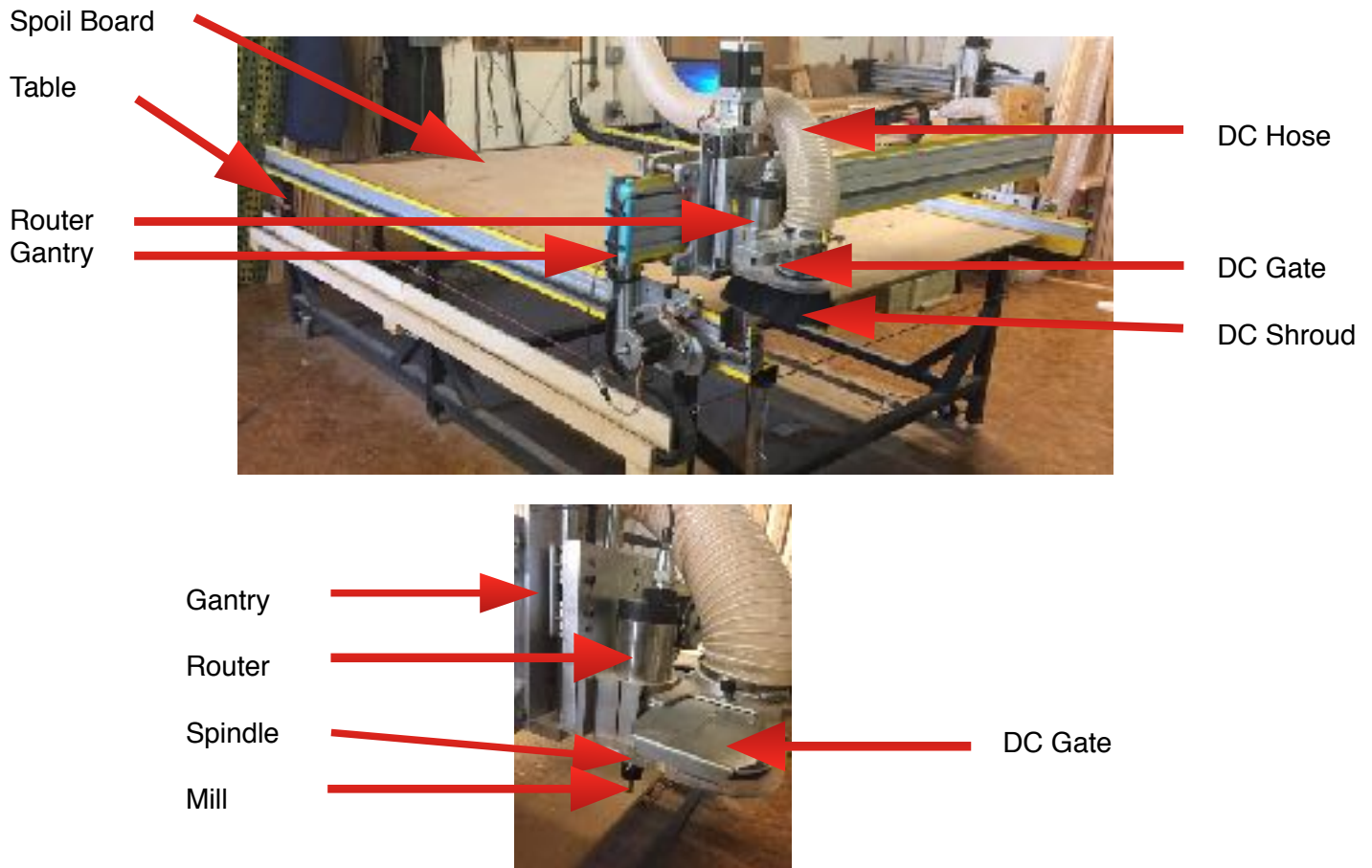
VCarve, Fusion 360, etc.

postprocessing (generate the text file that drives the CNC machine)

run a postprocessor to produce the text file (GCode) that will move the spindle to cut the toolpaths with the mills, feeds and speeds you specified in CAM

VCarve, Fusion 360, etc.

The Hive13-built CNC router has standard parts:
the **table** which supports the **spoil board**,
the **gantry** which moves the **router** in 3 axes in response to the **GCode** text file,
the **spindle**, which holds the cutting **mill** in the collet (compressible washer) and spins the mill,
the **dust collector** hose, **gate** and **shroud** (to...collect dust ;^)
the emergency stop cord (the purpose of the **E-cord** is left as an exercise for the reader...).



The CNC router works by moving the mill at a **feed** rate (GCode “**F**”)while spinning it at a **speed** rate(“**S**”), both of which are specified in the GCode text file of the **toolpath** along which it is moving. (VCarve GCode lines N150 & N190 specify the speed and the feed, respectively).

When not cutting, the mill is moved quickly between tool paths at an automatically set speed called the “**rapid**”. **It is important that your toolpath raises the mill be above all obstacles** (nail, screws, clamps) **when performing a rapid.**

Mills (similar to “router bits” in the woodshop) are specified by:

diameter (of cutter and also mounting shaft, if different),

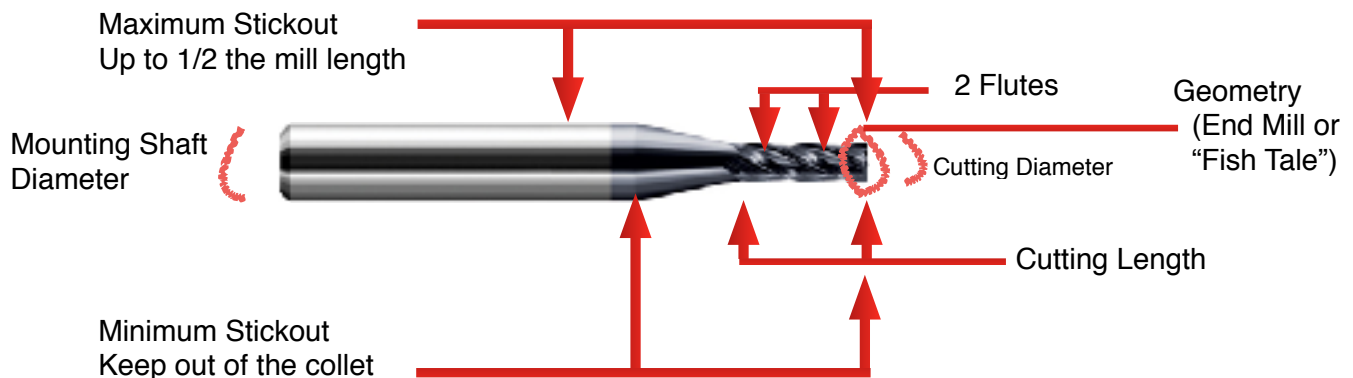
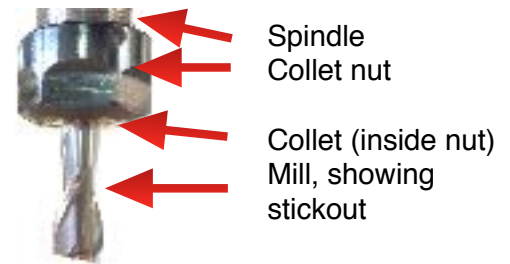
numbers of **flutes** (cutting edges, 1 or 2 for a CNC router),

geometry of the end of the cutter (end mill, ball nose, etc),

cutting length (length of the flutes),

stickout (the length sticking out beyond the collet - should not exceed 1/2 the total mill length)

and **material**: high speed steel (HSS) or carbide (you don’t need “coatings” for wood or plastic).



A starter set of mills could contain a couple of 3/16-3/8” end mills & a 60° v-bit with the same size shaft (so you won’t need to change collets when you change mills).

We **recommend against** using hand-held (or router-table) **router bits** in the CNC router. They are not built to tolerate the lateral stress of CNC machining and can break and send metal and cutters flying!

The only exception is hand-router **V-bits**. Although V-style **mills** are preferred, V-bits are considered safe in a CNC router when used for engraving lettering, as long as only a small part of the bit ($\leq 0.125''$ (3mm)) **cuts into the wood to engrave** the letters, minimizing lateral stresses. (i.e., the **Depth of Cut** is $\leq 0.125''$)

Mills are not drills. Mills are made to cut horizontally. **Don’t try to drill holes** with them. (Cutting out a **circle** that has a **larger diameter** than the mill is OK, it’s still a horizontal cut.) If you want to drill a hole, put a **drill bit** into the collet & use a “**peck drilling**” toolpath.

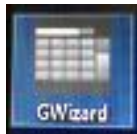
Taking a project from design to machine ready

As noted above, to calculate a toolpath, you need to have a feed, speed, and possibly a plunge rate. (The plunge rate is **not** for drilling, it determines how fast the mill can change depth while moving laterally.) When calculating Feeds & Speeds, you will need the **cutter diameter**, **number of flutes**, **stickout** and **material**.

Stickout was mentioned above as a range of values, i.e., “at least the cutting length, but no more than half the length of the mill.” The less the stickout, the less the vibration of the mill, and the cleaner the cut. It was also noted that you need the stickout to calculate the feed and speed. You need the feed and speed before you can run the mill on a toolpath, but you won’t know the actual stickout until you install the mill in the collet in the spindle. What?!

The answer is that, on a CNC router, you just need to get within about 1/8” of the stickout you used in your calculation when installing the mill. Install it and get as close to there top of the flutes as you can, without worrying too much. If you’re too far off, the mill may break, or the finish may be rougher than you expected. As with much of CNC routing, make an adjustment and try again.

Feeds & Speeds in the Woodshop



GWizard

(Note: GWizard is often updated and may be slightly different than described below. Check cnccookbook.com for info.)



Have the following info available:

material you’re cutting

tool: material (HSS or carbide), cutter diameter, # of flutes, stickout

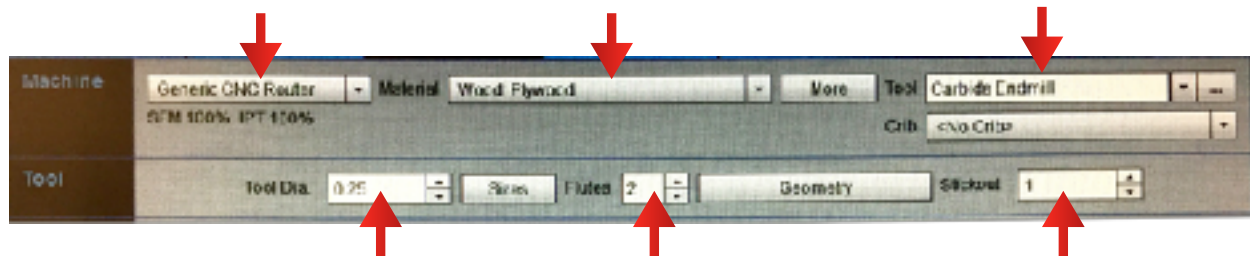
cut: depth & width(stepover)

Note: GWizard can be set to SAE or metric via the Setup tab - check that it matches your units!

Open GWizard; click on the Feeds/Speeds tab.



Machine From the drop-down menus, choose:
Generic CNC Router
 Material: **Wood** (hard, soft, MDF, plywood), Plastic or Aluminium, wax
 (no other materials on the this machine!)
 Tool: **Carbide** Endmill or **HSS** Endmill



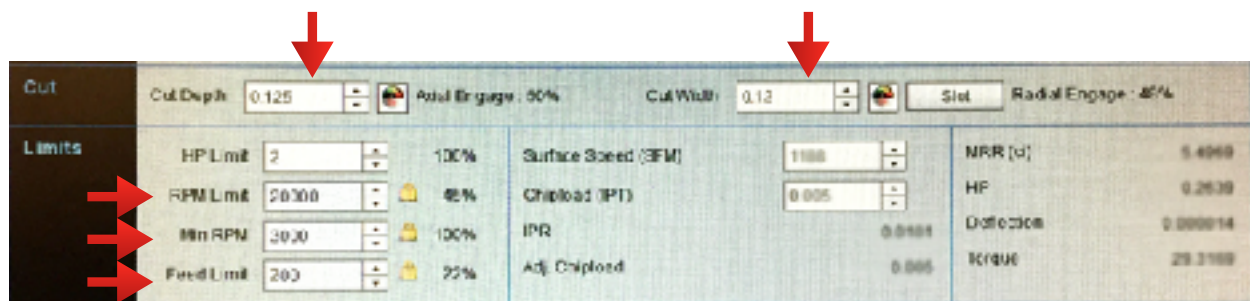
Tool Enter:
Tool Dia. (dia. of cutting end) (Check default units!)
Flutes: 1 for plastic or aluminium; 2 for wood and wood products
Stickout: (See above comments.) (Check default units!)

Mfg Skip for basic setup
 Mini Calc Skip for basic setup

Cut Enter:
 Cut Depth: wood: start with the radius of the cutting end (“Axial Engage” 50%)
 plastic or aluminium start with **1/5** the **radius** and see how it goes
 Cut Width: less than the radius of the cutting end (“Radial Engage” 12.5% - 48%)
 (“Cut Width” is often called “stepover”. It is the width of the cuts which occur after the first cut.)

Limits (Verify or enter the following value to match the default setup for our CNC router.)

RPM Limit: **20000**
 Min RPM: **0**
 Feed Limit: **200** (Skip the rest for basic setup)



HSM Skip for basic setup

Tips

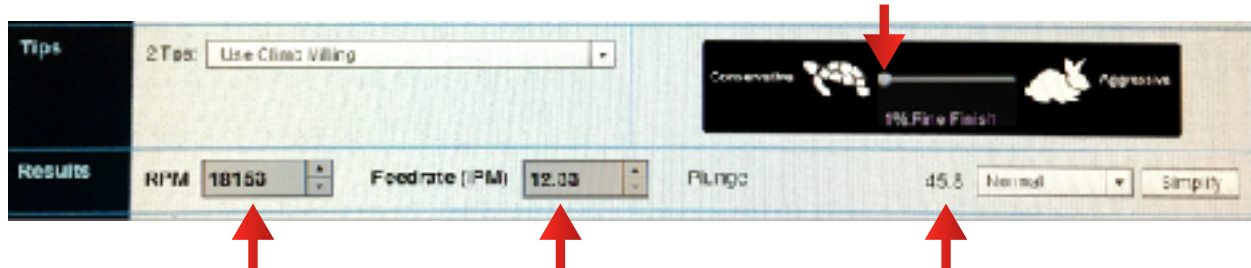
May have some interesting suggestions



(Tortoise - Hare panel)

(Click on the slider & adjust with the mouse or arrow keys between 1% and 100%.)

Adjust the % until one or both numbers (RPM and Feedrate) turn red,
then back off until both are black again.



This will show you the calculated max and min RPM (“Speed”) &
Feedrate (“Feed”)(in Inches per Minute, IPM).

It may also show a Plunge rate.

The results at 29% are estimated by the software as giving the optimal
compromise for maximum Feedrate still resulting in a smooth finish.

This is just a guideline. YMMV.

If the RPM (“Speed”) stays at 20,000 (in the red) even at the lowest Feedrate, adjust the panel %
and find the max & min Feedrate that appear in black. Use these as your limits at 20,000 rpm.

You will use this data in the “Edit” panel after you “Select” a tool (and click **OK**)
in a VCarve toolpath. (see below)

The higher the Feed, the faster, but rougher the cut.

Play with the Feed (within the limits you found in GWizard) and see what gives satisfactory results on the
material you’re using.

Results

(Write these down for use in VCarve.)

RPM (“Speed”)

Feedrate (IPM) (“Feed”)

Plunge

(You will also need the cutter diameter and stickout you used above for use in VCarve.)

Cut KB

skip for basic setup

CAD / CAM (short intro. Much more info is on the Vectric website.)

Launch VCarve Pro 9

(Download the VCarve Pro trial version and enter your makerspace registration.)

If you are just postprocessing the toolpaths in your pre-saved project file (.crv) to generate the GCode, go to **Postprocessing**, below.

If you are resuming a prior project, click **Open an existing file.**

If you are starting a new project, including importing a design from, say, Inkscape, click **Create a new file** to fill out the **Job Setup** panel.

Create a new file - this will open the **Job Setup** panel.

Check **Single Sided**

(You can do 3D machining on one side, or on two sides. See tutorials on the Vectric website.)

Enter **Width** (x) and **Height** (y).

and **Thickness** (z) of material.

(VCarve will calculate fractions for you: e.g. enter “18(space)1/16=” the result is 18.0625)

Verify your **Units**.

Set **Z Zero** to **Machine Bed**.

Set the **XY Datum Position** to the lower left corner of your material. (It helps to mark this on your material.)

Click **OK** - this will open the **Drawing** panel.

If you need to get back to the **Job Setup** panel, click on the white square icon in the second row under **File Operations**.

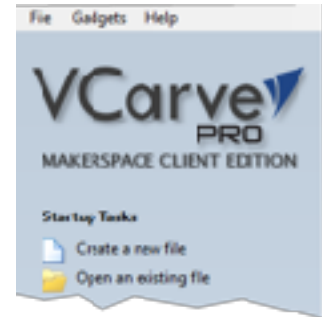
CAD (Computer Aided Design)

Mark the placement of the hold downs (nails, etc.) with 1.5" circles using the circle icon under **Create Vectors**.

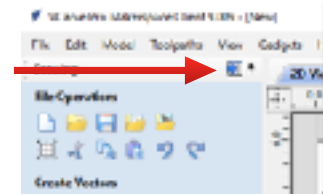
Import (under **File**, in the menu bar) or create the vectors that will define your design.

If you import vectors, check for redundant or open vectors. Eliminate the former and close the latter. See the Vectric website for vector editing info.

Use **Offset** under **Offset and Layout** to make a path for cutting your project out of the material, if needed. Offset it by the radius of the mill you will use.

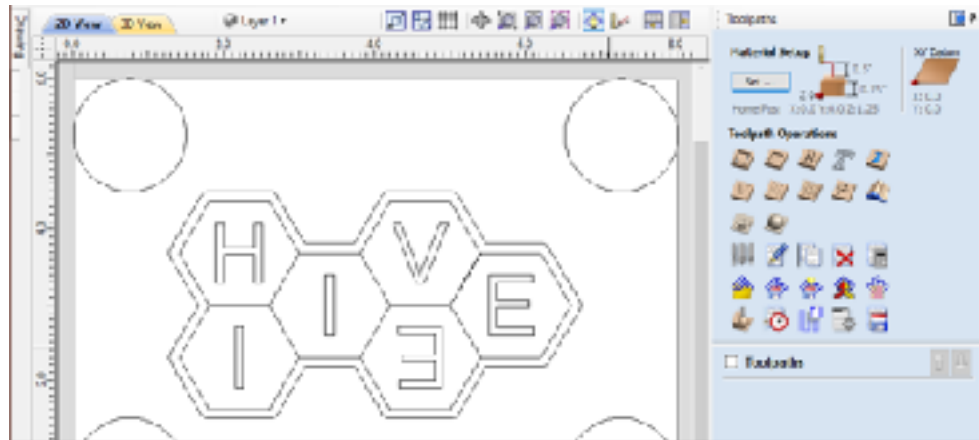


Click the right-pointing blue arrow icon (**Switch to Toolpath commands**) at the top of the **Drawing** panel to get to the **Toolpaths** panel.



CAM (Computer Aided Manufacturing (toolpaths))

Before creating toolpaths, we need to set up our parameters for the GCode.



Under **Material Setup** in the **Toolpaths** panel, click **Set...**
This will open the **Material Setup** panel.

Check that the **Thickness**, **XY Datum** corner, and **Z-Zero** location match your settings in the **Job Setup** tab.

In **Rapid Z Gaps above Material**, enter values for **Clearance(Z1)** and **Plunge(Z2)**

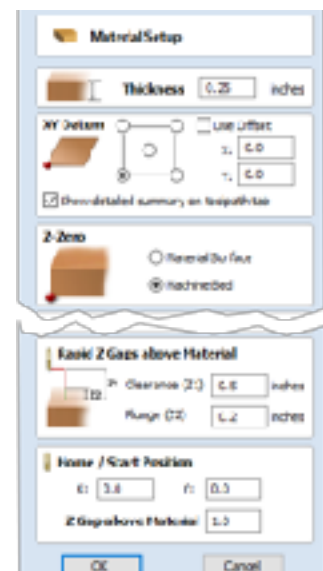
Z1 to clear nails & screws, usu 0.5-1" (or above any clamps!)

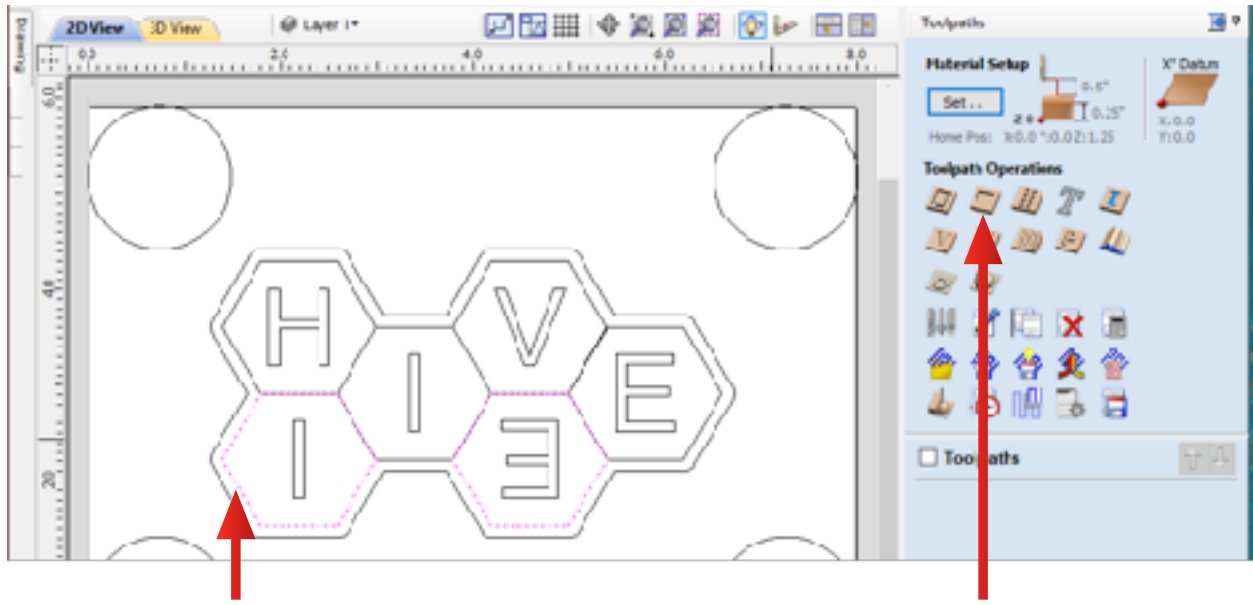
Z2 to prevent the descending mill from crashing into material, 0.2"

In **Home/Start Position**, enter values for **Z Gap above Material**

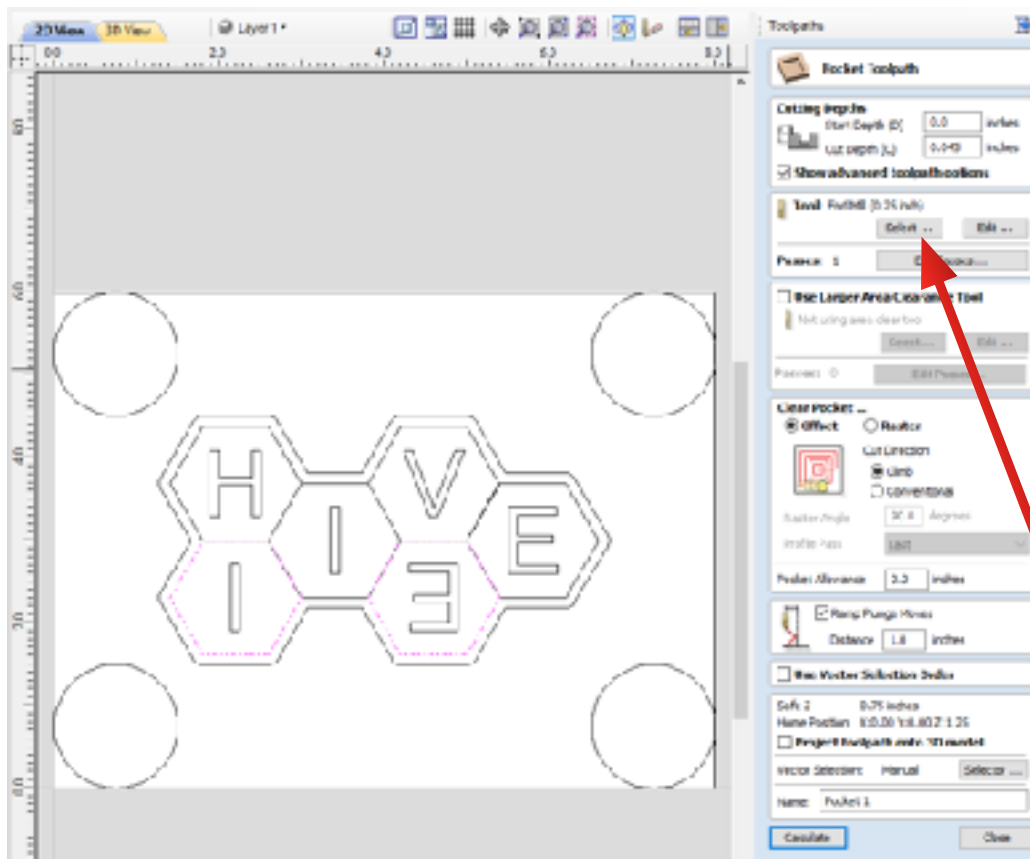
Calculate as **Z Gap above Material** as **Thickness + Z1 + Z2**
(We rounded up from 0.95" to 1.0")

Click **OK**. This will take you back to the **Toolpaths** panel.





Choose each vector in your design in turn and click on a toolpath icon (here a **Pocket** toolpath).

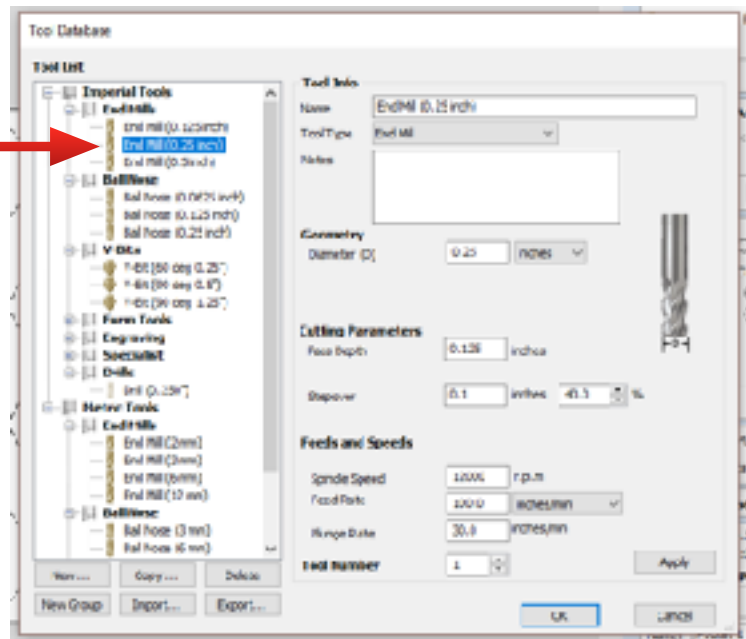


Define the **Start depth**.
(0 = the top of your material.)

Here we set a depth for the pocket we are cutting, of 0.045"

Choose a tool type from the crib via **Tool: Select**

Choose a tool.



Click **OK**.

Specify the parameters for *your* mill via **Tool: Edit...** (next to **Tool: Select**, above)

Verify that **Diameter** matches the cutting diameter of your tool.

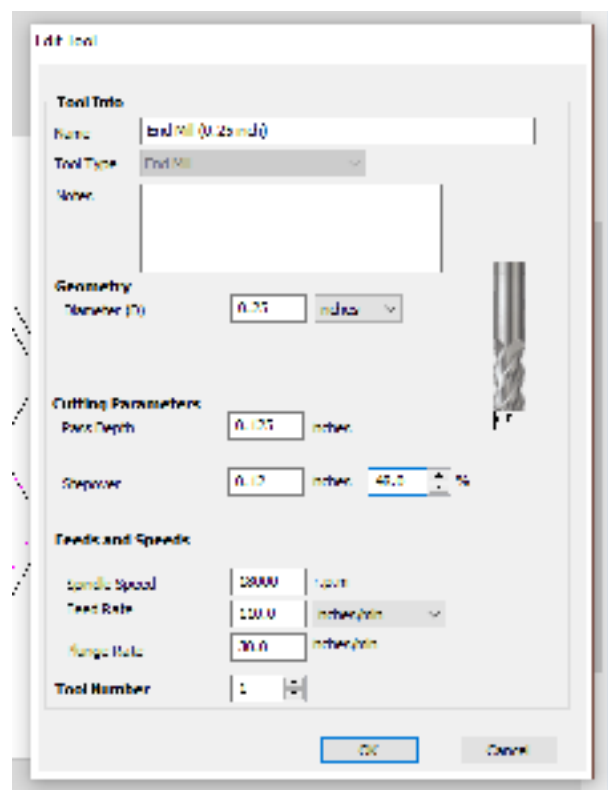
Start (in wood or plastic) with a **Pass Depth** that is no greater than the cutter radius.
(1/5 of the radius for aluminium.)

Stepover should be less than the **Pass Depth**.

Enter the **Speed**, **Feed** and (if applicable) **Plunge** for your tool.

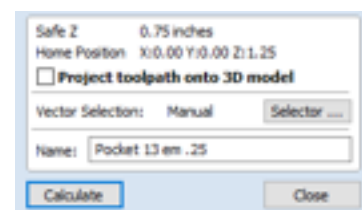
Be sure the units for **Feed Rate** are correct (**inches/min**).

Click **OK**.

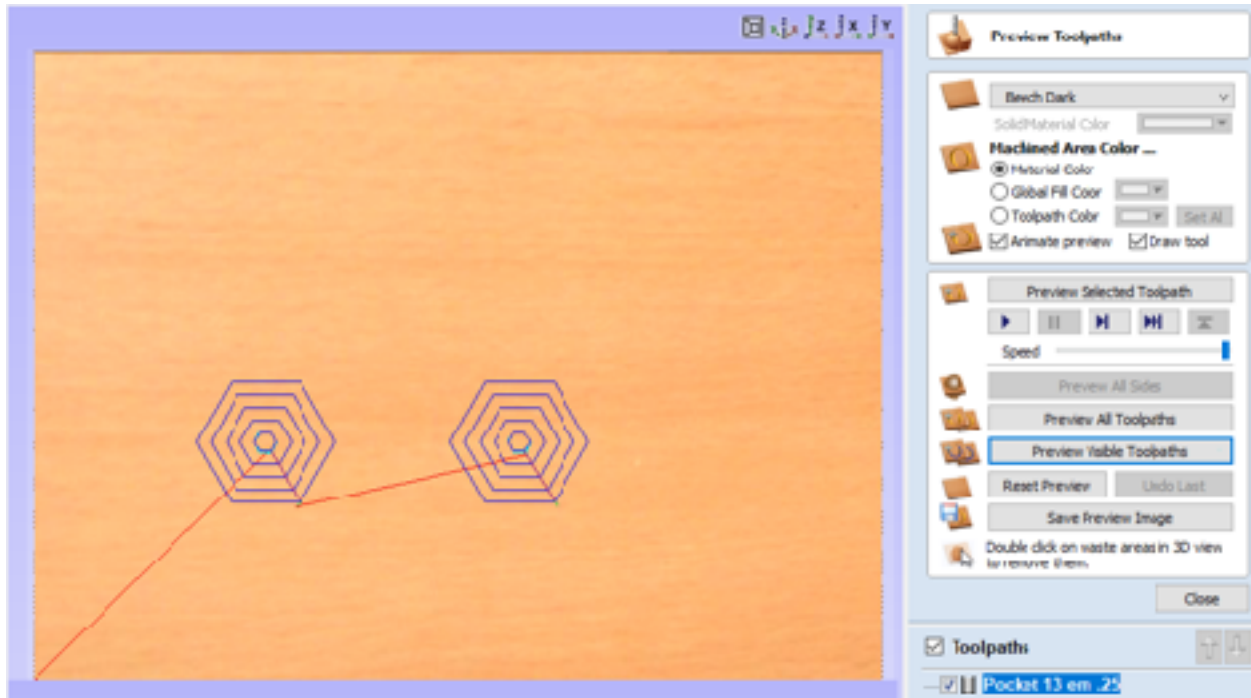


Name the toolpath so you know what project, what vector and what mill (type & size) are used.

Click **Calculate**.



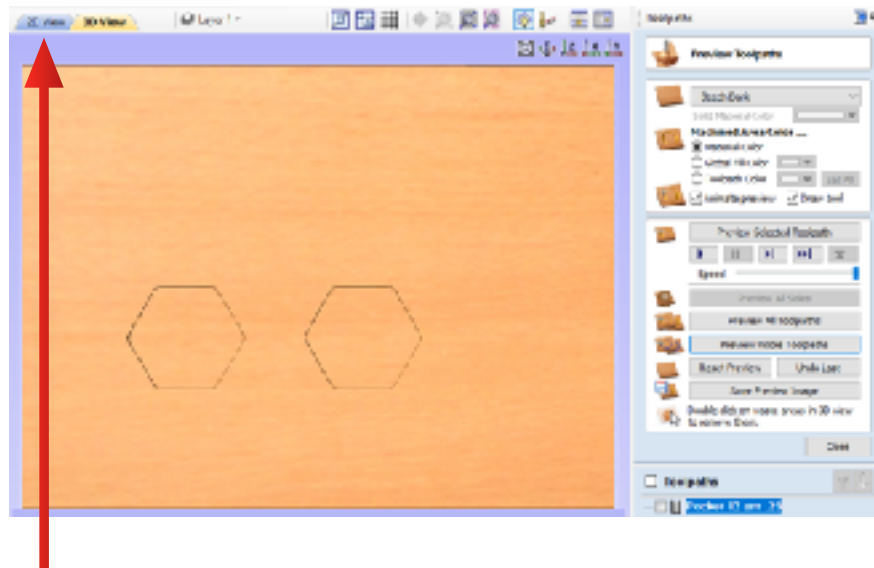
This will open the **Preview Toolpaths** panel and the **3D View** (simulator) window.



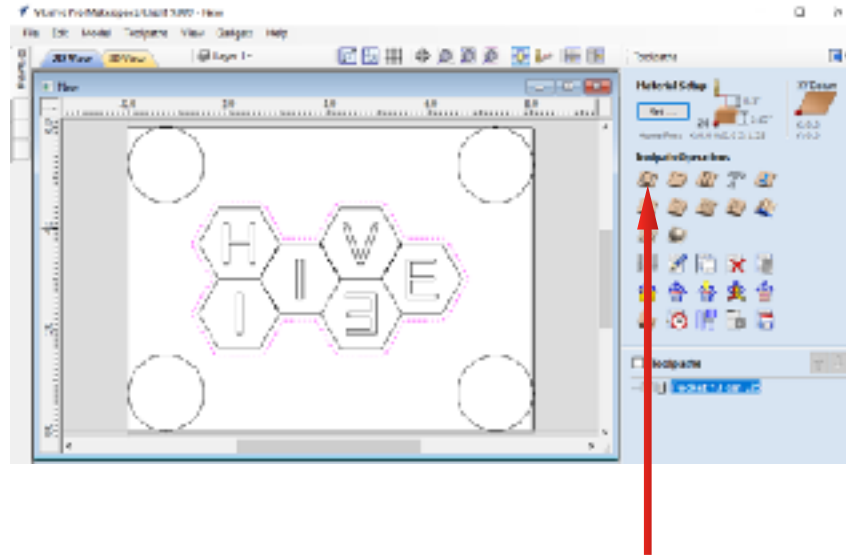
Be sure there is a check in the box in front of the tool path you want to simulate.
Click **Preview Visible Toolpaths**.

VCarve will show the results of the simulation of your toolpath, in the **3D View** window.

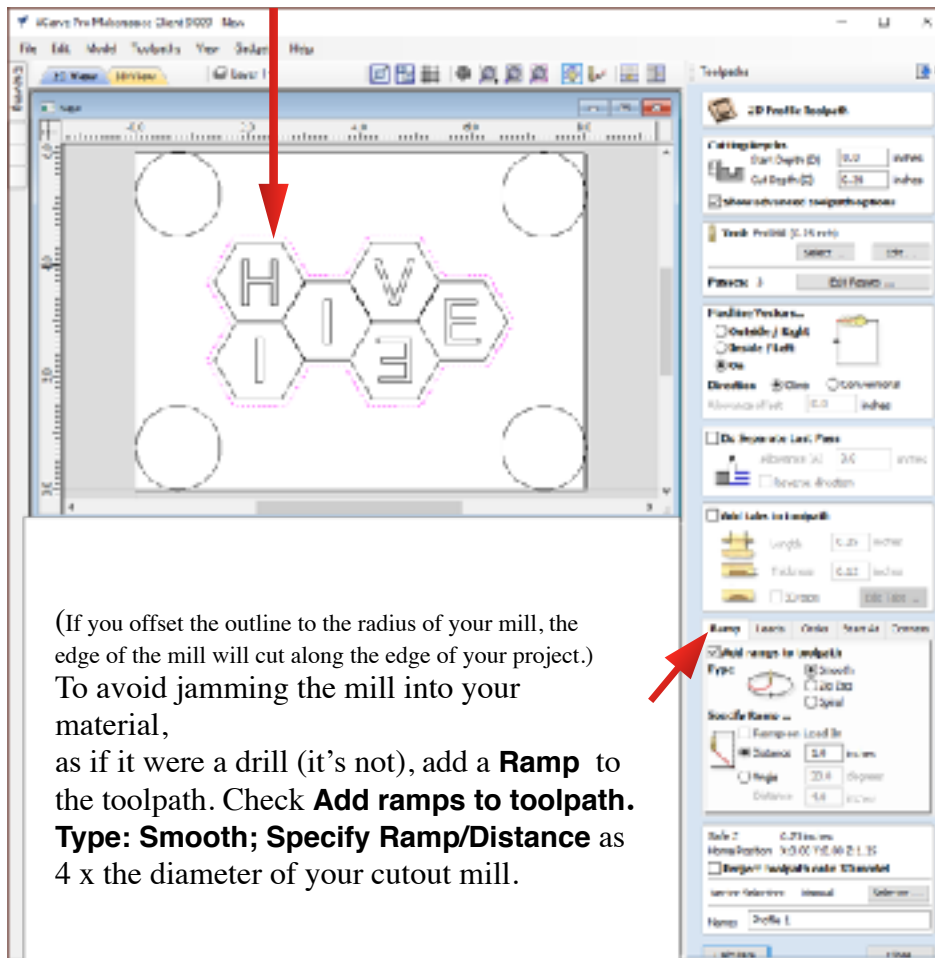
To set up the next toolpath, click **Close**.
You will go back to the **Toolpaths** panel.



To re-open the vector display, click on 2D View, at the upper left.



If you are cutting your project out of the blank, highlight the **Profile** tool path and the **Offset** vector you used to outline your project above.



(If you offset the outline to the radius of your mill, the edge of the mill will cut along the edge of your project.) To avoid jamming the mill into your material, as if it were a drill (it's not), add a **Ramp** to the toolpath. Check **Add ramps to toolpath**. **Type: Smooth; Specify Ramp/Distance** as 4 x the diameter of your cutout mill.

Set the **Start Depth** to 0.
Set the **Cut Depth** to material thickness + 0.02" to 0.04"
(Type "z=" then "0.04+")
Select a tool.
Click **OK**.

Edit the tool.

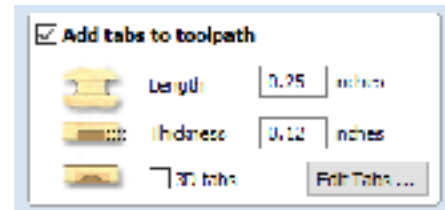
Under **Machine Vectors**, select **On**.

Finally, to prevent the project from flying off the table as it is cutout, add tabs to hold it in place.

In the sub-panel above **Ramps**, check **Add tabs to toolpath**.

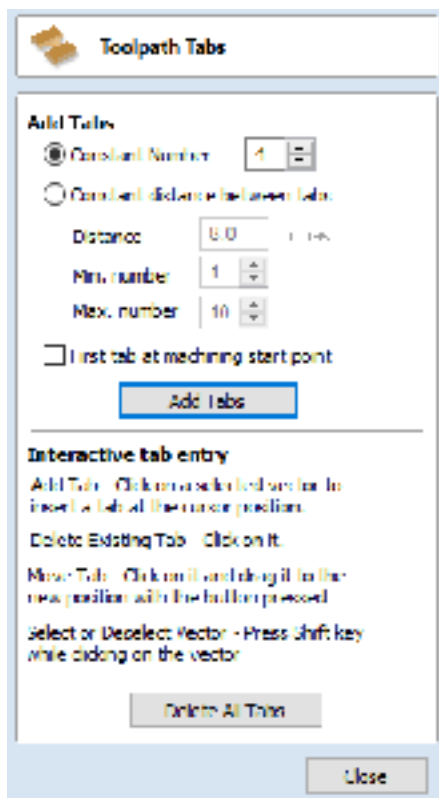
A good place to start is a **Length** of **0.25"** and a **Thickness** of **0.12 - 0.25"**

Click **Edit Tabs...**



This will open the **Toolpath Tabs** panel.

You should put some along each edge, about every 6"-12.

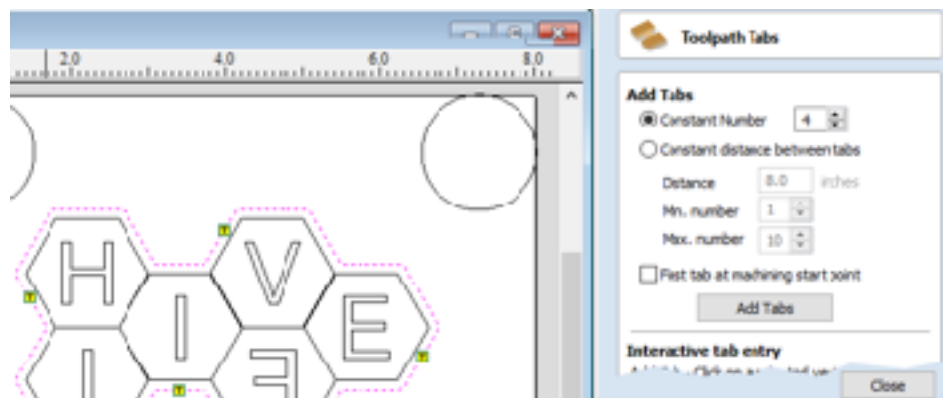


Using the "+" part of the cursor, click the vector to which you wish to add the tabs. A tab will appear where you click.

Small yellow squares will appear along the highlighted toolpath.

You can move the tabs with the cursor, or click on them to erase tabs.

Click **Close**.

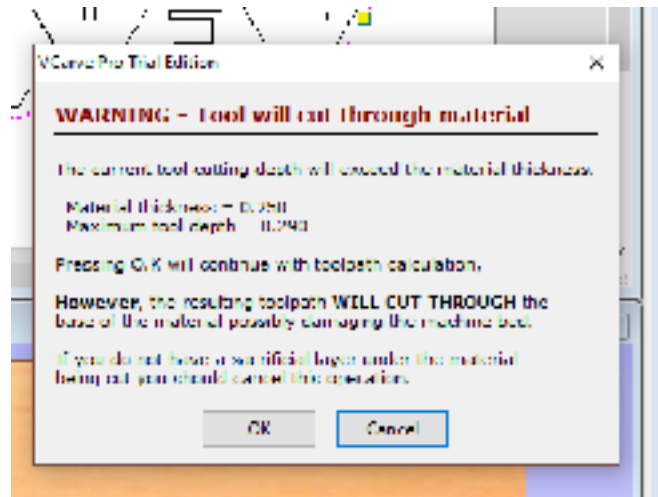


Click **Calculate**.

Be sure you get a **WARNING** from VCarve that you are going to cut through the material.

If you do not see this warning, your project will not be cut from the surrounding material.

Check that the **Maximum tool depth** is no more than **0.04"** (1mm) greater than the **Material thickness**.



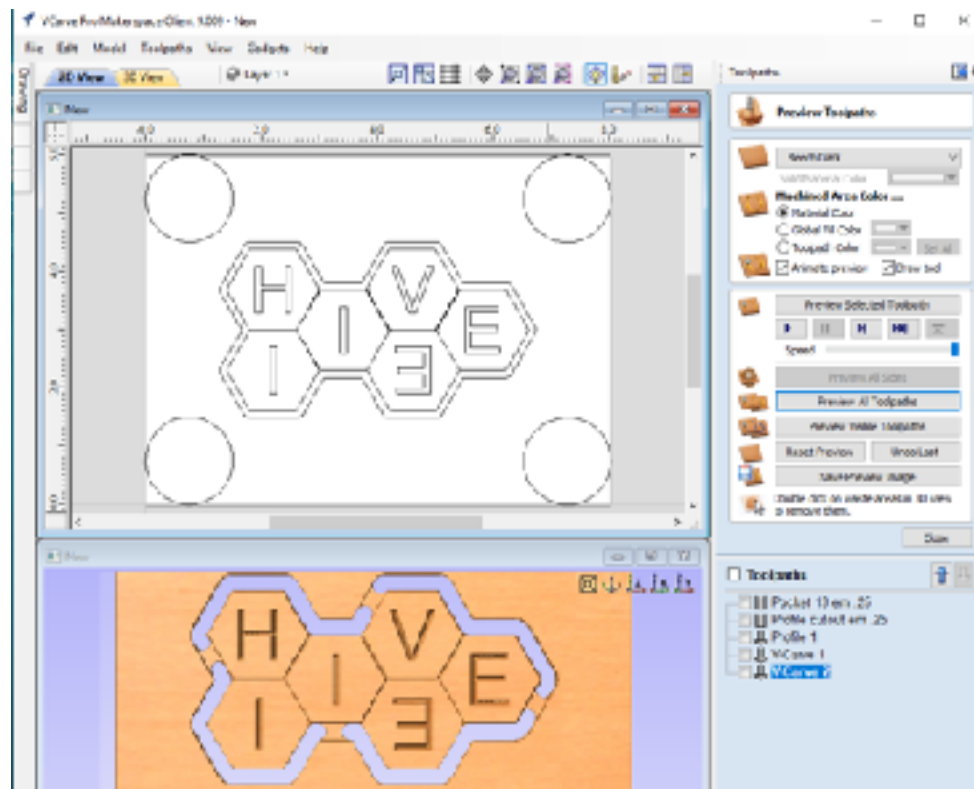
Click **OK** for the warning.

The toolpath will be calculated.

and this will open the **Preview Toolpaths** panel and the **3D View** (simulator) window.

To see the results of all your toolpaths at once,

Click **Preview All Toolpaths**.



After all tool paths are calculated & checked, **Save** the file via the **File** menu. VCarve will save the file in the .crv format.

If you try to **Save Toolpaths** now, the software will refuse.

Take the .crv file you saved (e.g., to a USB drive) to Hive 13 and upload it to one of the computers running a registered copy of **VCave Pro 9**.



Postprocessing

At Hive 13, launch VCarve Pro 9.

Open an Existing File to import your file into VCarve Pro 9.

When asked, affirm that you want to register (or re-register) it.

Go to the **CAM** window (**Toolpaths** panel), as above.

Click the floppy disk(citation needed) icon to open the **Save Toolpaths** panel.

Be sure that **Mach2/3 Arcs (inch) (*.txt)** is the **Post Processor** being used.

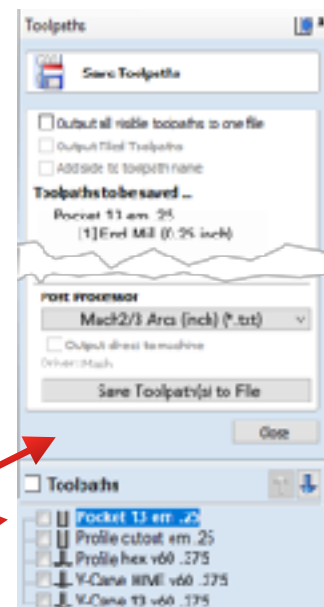
Highlight each toolpath name in turn.

Click **Save Toolpath(s) to File**

for each toolpath.

You can save them to a USB drive or put them in a folder in the Z: Drive, DUMPSTOR

These are the GCode text files that you will load into **Mach3** to machine your project.



Set Up and Run Your Project

Be sure you marked the **X,Y Datum** corner & **Z Zero** (top or bottom of your material).

Be sure you know the **safe hold-down** (nail/screw/clamp) locations.

Use your eye and ear protection!

Set Up the CNC Router

Power up: PC on

Boot **Mach3**



Select the most recent **Profile** in the popup window.

Click **OK**

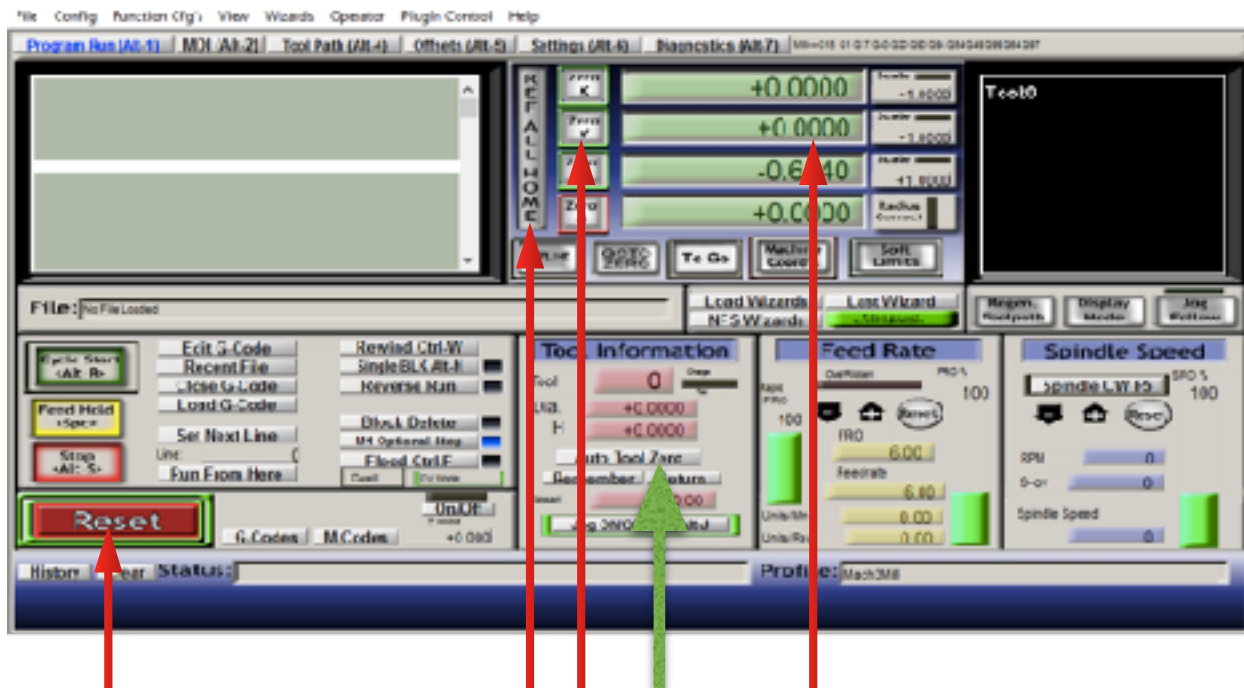
(It will complain that it can't see the router. Once you turn the router on (below), click on "Retry", Mach3 will now launch.)

Flip the **white switch** on the wall to power up the CNC router.



When "**0000.0**" appears on the spindle driver display, press **RUN**.

Zero the Machine X & Y:



In **Mach3**: Double-Click **Reset** to show a steady green border.
 Click **REF ALL HOME**. The router will go to table X,Y = 0,0.
 The borders of **Zero X**, **Zero Y** and **Zero Z** will change to green.
 Click **Reset** to show a blinking red/green border.
 Click **Reset** to show a steady green border
 Click **Zero X**. Click **Zero Y**. The numeric displays will reset to 0.
 The machine is zeroed to the table **X** zero and **Y** zero ("machine coordinates").

Zero the Machine to the first mill stickout (Z0):

Use **Page Up** on the keyboard to move the Z axis up about 2".
 Use the arrow keys on the keyboard to move the spindle about +6" on X and on Y axes.
 Click **Reset** to show a blinking red/green border. (This locks the spindle.)



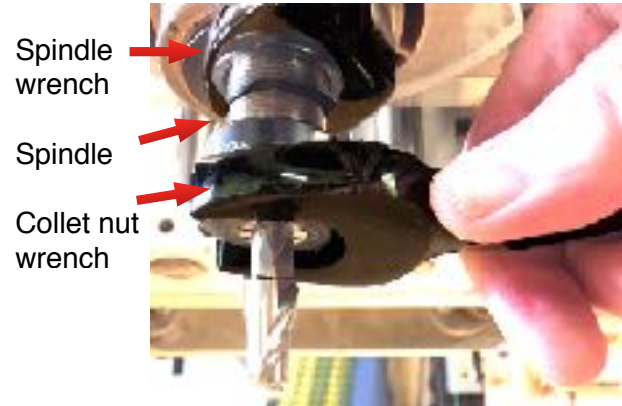
Collet



Install the collet: place the nut on the spoil board and snap the collet in.
Finger tighten the assembly into the spindle (CCW, seen from above).
 Install the 1st mill: press the mill into collet (keep the flutes outside the collet); **finger** tighten.

Hold the wrenches as **close to the spindle** as possible & **tighten** the collet nut using only **one hand**. It should only rotate **about 1/4 turn (CCW** from above) to firmly secure the mill.

Put the wrenches back on the hooks,
don't leave them on the spoil board!



Do Not Overtighten!

this will **destroy the threads**
and render the **\$600 Spindle useless!**

Connect the Z Zero probe.

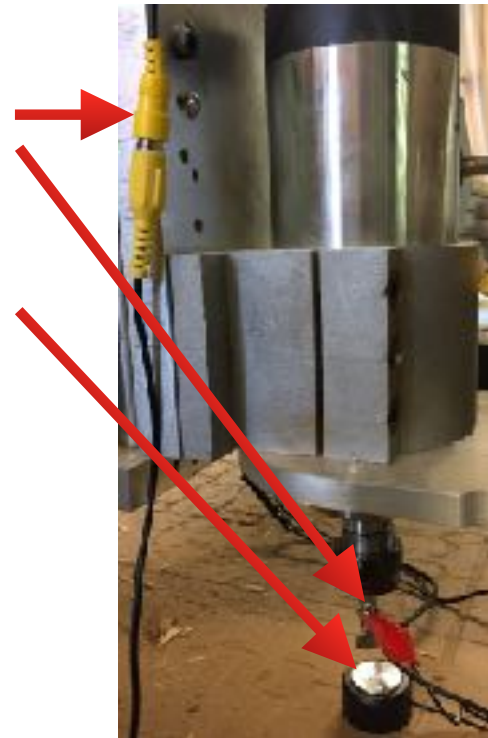
Connect the yellow RCA jacks.
Clip the red-shrouded alligator clip to the mill.
(Watch that it doesn't disconnect while the router is moving to touch the contact button.)

Place the contact button on the spoil board.

Click **Reset** to show a steady green border.
Click **Auto Tool Zero**. (green arrow, fig. above)
The router will descend until the mill touches the contact button.
It will then rise to 1.5" above the spoil board.

After the spindle touches probe & **rises to 1.5"**,
remove the probe.

The machine is now zeroed to this mill stickout (Z0).



Set Up Your Material (zero to your material X0 & Y0)

Place your material on the spoil board; secure it square to the machine rails.

Center your mill on your project **X,Y Datum** corner.

Click **Zero X**. Click **Zero Y**.

This tells the machine where your **project/material XY Datum Corner**,
("Home X,Y") is. All toolpaths start here. ("Project coordinates")

Open the Dust Collector (DC) gate on the router.

Check that the other gates in the shop are closed for maximum suction.

Run Your Project (zero to the mill Z0 *each time you change mills*)

(—>) Click **Reset** to show a blinking red/green border.

Change the mill if necessary.

Click **Reset** to show a steady green border.

Page Up to put the mill 2" **above** your material.

Use the arrow keys to move the mill for zeroing the in the Z axis.

Connect the Z Zero probe.

If Z zero set to the base of material, place the button on the spoil board.

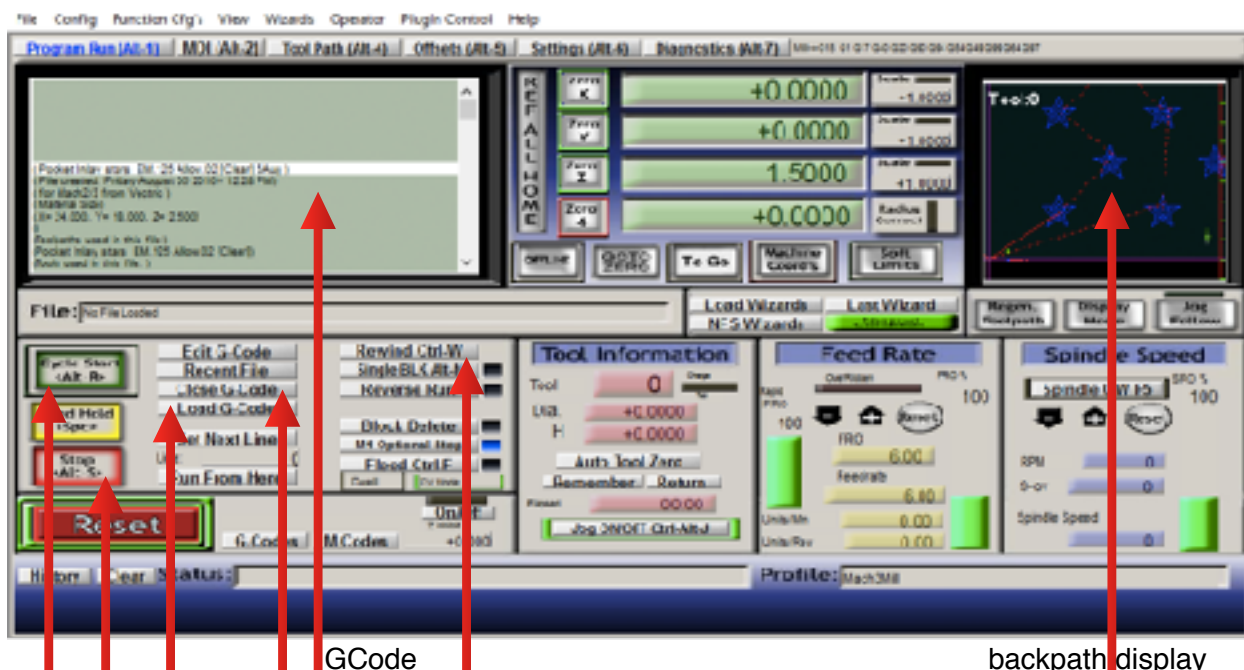
If set to the top of the material, place the Z button on the material.

Click **Auto Tool Zero.**

Remove probe once the machine has raised the mill to 1.5" and stopped.

Replace the dust collector collar.

Turn the **Dust Collector on!**



Click **Load GCode**

Check that the backpath window shows the expected toolpath pattern.

Scroll through the GCode for tool path name, installed mill,

speed (≤ 20000) (line N150nnnnnn**Sxxxxx**) and

feed rate (≤ 200) (line N190nnnnnnnn**Fxxx.x**).

Click **Rewind**. Failure to click **Rewind** could result in damage to your project!

(->>) Click **Cycle Start**

Check that the mill is spinning @ Speed!

(Look at the spindle driver display readout. Note: it may be up to about 10% off.)

Do Not Leave the Machine While It's Running!

Be ready to click **Stop** or **Reset!**

If you have to **Stop** the machine before the tool path is complete:

Raise the mill above your material

Correct the problem (You may need VCarve to change the toolpath.).

If you changed the toolpath, Go to (—>)

Otherwise: click **Rewind**. Go to(->>).

When tool path is completed, **move the mill away** from the material.

Click **Close GCode**. (fig. above)

Remove the dust collector collar.

Go to (—>).

What could possibly go wrong?

or

Do Not Leave the Machine While It's Running!

Problem: Ref All Home does not send the router the the lower left corner **or** the **machine tries to push past the stops**.(I.e., the machine won't auto-zero to the table.)

Response: Stop!

Cause: Limit sensors are off line.

Fix: Quit Mach3, shut off the PC, shut off the router, restart as above (PC, Mach3, router)

(If **Ref All Home** fails again, move the gantry to the **XY Datum** corner of your material. Click **Zero X & Zero Y**)

Problem: Auto Tool Zero doesn't finish by retracting to 1.5" above the Z Zero probe.

(I.e., the machine will not auto-zero to the mill stick-out.)

Response: Stop! Raise the mill off the button and remove the probe.

Cause: Limit sensor is off line.

Fix: Quit Mach3, shut off the PC, shut off the router, restart as above (PC, Mach3, router)

(If **Auto Tool Zero** fails again, move the mill to a spot where you can touch it to your Zero Z plane (the spoil board or the top of your material).

Bring the mill down to about 1/4" above the Z Zero plane using **Page Down**.

Put a regular (not thick) sheet of paper under the mill.

Lightly tap the **Page Down** key repeatedly, until the paper just fails to slide out from under the mill.

Click to highlight the green panel next to **Zero Z** in Mach3.

Type in **0.003** (the thickness of the paper), and press **Enter**.

The panel will reset to 0.003.

The router is zeroed to the current mill stickout, i.e., **Z Zero**

The above procedures will zero the machine to your material and your current mill.)

Problem: The router plunges through your material and cuts directly into the spoil board.

Response: Stop! Raise the mill!

Cause: You **zeroed the mill Z** to the **top** of your material in **VCarve**, but to the **spoil board** at the **CNC router**.

Fix: Zero the mill **Z** to the proper surface. Replace the material. **Rewind** the GCode and restart the toolpath.

Problem: The mill breaks.

Response: Stop! Raise the mill!

Cause: A break usually means either the feed is **too fast** or speed is **too slow** or you are cutting too greedily & **too deep**.

Check that you didn't crash into a screw or clamp.

Check that you didn't hit a knot if you are cutting wood.

Fix: Correct feed, speed, path, pass depth (or number of passes) or material!

Replace mill. Zero to mill stickout. Rewind the GCode & restart the toolpath.

Problem: You smell or see smoke.

Response: Stop! Raise the mill! Move it away!

If there is **fire**, **RACE** (see below)!

If there is **smoke**, clear the chips from the path, but keep them on the table.

If the smoke persists, check the dust collector box. **RACE!**

Cause: Rubbing (spinning without cutting): feed and/or speed is **too slow** or **chips are not being cleared** fast enough.

Fix: If the smoke clears when you move the chips, use the ShopVac® to manually assist the DC as it clears chips while the toolpaths are cutting.

Rewind the GCode and restart the toolpath.

Fire Safety - RACE

Rescue others & yourself - yell "Fire! Tell others to clear out; leave.

Alarm Call 911 - from outside if it's a large fire!

Stay to talk to Fire Dept. & report any missing persons

Contain if it's a large fire; close the doors as you leave.

Extinguish if small (and you've trained with an extinguisher)

or

Exit if the fire is large.

Catch These Problems Early: Do Not Leave the Machine While It's Running!

Clean Up Any Mess!

When the project is complete, chop the tabs (mallet & chisel) and remove your project.

Remove your material.

Remove all hold-downs.

To leave a smooth spoil board for the next person, pull out or sand down any plastic nail remnants sticking out of the spoil board.

Clean up the chips on the table and **floor!**

*Please do **not** use an air gun to blow chips off the spoil board!*

*Metal shop chips fall to the floor. **Wood chips & sawdust float fer-ever!***

Return the collet to the tray and the collet nut to the left table rail, near the wrenches.

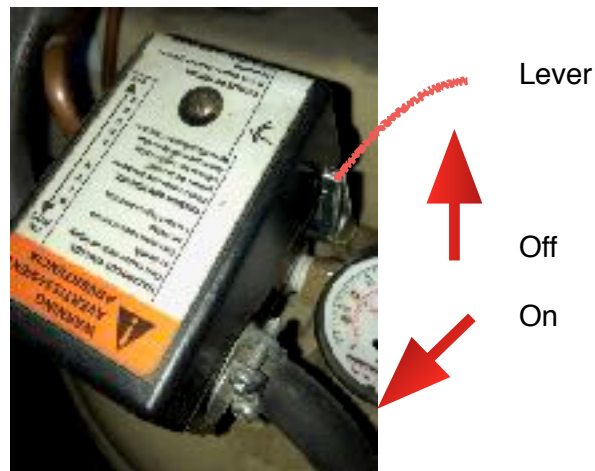
Quit **Mach3**.

Power down the router (white switch), **dust collector** and PC.

Empty the DC box, if needed!

Please remember to **power down the air compressor** (in the back left corner of the metal shop)!

(The lever points ~ up for off,
~ 30° below horizontal down for on.)



To be **certified** to use the Hive 1 3 CNC router, you will need to take the Hive 1 3 CNC Intro Course and then arrange for a Hive 1 3 **CNC machinist** to observe you running a project. Post your proposed time & date on the hive13.org website mailing list page to ask for coverage.

Acknowledgements

Screen shots for Feeds and Speeds... from GWizard Software by CNCCookbook; download at <https://www.cnccookbook.com/index>

Screen shots for CAD / CAM from Vectric's VCarve; download at vectric.com

Screen shots for Set Up and Run Your Project from Newfangled Solutions' Mach3; download at machsupport.com

References

CAD/CAM/postprocessing



VCarve vectric.com - downloads, FAQs, tutorials

Fusion360 autodesk.com/products/fusion-360/overview (if generating GCode in Fusion, *de-select* G28)

GWizard

CNCCookbook cnccookbook.com/GWizardGWizardInstall1SC.html (& info on all aspects of CNC milling)

GCode

Tormach http://www.tormach.com/machine_codes_gcodes.html

descriptions of GCode “words” (commands) like the “S” word for speed and the “F” word for...feed rate.

End mill info (no endorsement by our facility or guarantee of quality / results)

Intro to end mills: <http://makezine.com/2014/09/10/endmills/>

Basic assortment (“carbide” lasts longer):

1/2” & 1/4” 2 flute upcut end mills (for profiles & pockets in hardwood and plywood)

(also: 1 flute for Al or plastic, downcut or compression for plywood or laminates)

1/2” 60° v-bit (for engraving lettering, etc.)

1/4” 2 flute ball nose mill (for 3D contours)

Sales: (local but pricy) Rockler, Woodcraft, McMaster-Carr, Fastenal

Online: [kodiakcuttingtools](http://kodiakcuttingtools.com), [toolstoday](http://toolstoday.com), Eagle America, Southwest Tools, Whiteside, Amana (via Amazon), MSCDirect